FADAL MACHINING CENTERS

GE FANUC OPERATOR MANUAL
18i MB5 Digital CNC

FADAL MACHINING CENTERS, LLC

TABLE OF CONTENTS

1.0 PRESTART CHECKING STEPS

1.1 OIL RESERVOIR

Examine the oil levels. Both levels should be filled up to one inch from the bottom of the reservoir. The spindle oil reservoir may have oil in it for up to six months. The way lube oil reservoir may run out of oil in one week.

☞ **NOTE:**
VMCs with linear way systems require grease. See the Maintenance Manual on specifications on the way lube and the spindle oil.

1.2 AIR PRESSURE

Visually inspect the air pressure gauge to verify that it is set to at least 80-100 PSI. Air is used to change belt ranges in the spindle, orient the spindle, activate the tool in-out cylinder, and it is, also, used for the air blast during a tool change. The tool changes gauge should not exceed 120 PSI.

1.3 WATER RESERVOIR

Most new VMC models release water collected in the water reservoir automatically. It is advisable to place an additional water trap in the air line going to the machine.

1.4 FLOOD COOLANT

Replenish the flood coolant level to avoid running out of coolant during execution of the program.

1.5 SPINDLE COOLER RESERVOIR

Examine the spindle cooler reservoir once a month.

2.0 OPERATOR PANEL LAYOUT

2.1 POWER ON/OFF

2.1.1 POWER ON

1. Turn On the main breaker
To power on the machine, press the safety lock and turn the power switch in the clockwise direction.

2. Press the CNC ON push button. CNC will boot up and enter operating status.

2.1.2 POWER OFF

Operator Panel B layout

1. Press the CNC OFF push button.

2. Turn off the main breaker.

To power off the machine, turn the power switch in the counterclockwise direction.

AUTO	EDIT	MDI	REMOTE
SINGLE BLOCK	BLOCK DELETE	OPT STOP	WORK MEAS
TOOL MEAS	MC LOCK	DRY RUN	TOOL BRKN
DOOR OVRD	DRUM FWD	DRUM REV	WORK LIGHT
CYCLE STOP	CYCLE START	PRG STOP	ALARM MSG

REF RETUR	JOG	INC JOG	HANDLE
X 1	X 10	X 100	X 1000
HOME PSN	LOAD PSN		Z HOME
DRUM PSN	WASH DWN	INIT MAG	TOOL DATA
TOOL REL	TOOL CHNG	LOAD TOOL	FUNC

MIST COOL	FLOOD COOL	AUTO COOL
X	Y	Z
4	5	6
+	TRVRS	-
SPDL CW	SPDL STOP	SPDL CCW

GE Fanuc 18i-MB5 control is equipped with "Main B" Operator Panel. 55 Push button layout is pictured above.

Clear key caps and custom legends have been installed as presently defined for machine use. Unless otherwise noted, buttons fit with blank key caps (no legend) are not defined for user-customer use.

2.2 SAFETY FUNC-TIONS

2.2.1 EMERGENCY STOP SWITCH
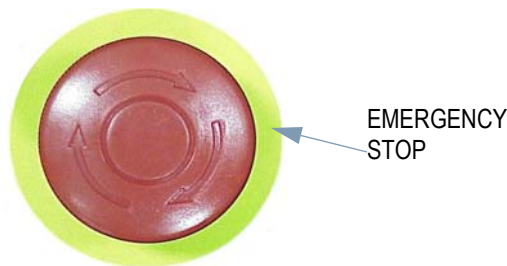
EMERGENCY STOP

If you press Emergency Stop button on the machine operator's manual, machine movement stops in a moment. This button is locked when it is pressed, and can be unlocked by twisting it.

When the emergency stop is pressed, the emergency stop command is applied to the machine, and the CNC is reset, spindle and XYZ axes are stopped, and the other actions are interrupted. CNC will display EMG STOP message on the screen.

2.2.2 DOOR OVRD

This push button enables opening machine doors without generating Feed Hold & Cycle Start inhibits. While door override is applied, LED is flashing & Operator Message (2006 Door open override is active) is displayed to remind that override is active. Operation of Door Override push button may be selected as latching (default) or momentary style by user. Latching operation provides On/Off toggle function. Momentary function requires operator to hold push button while door override is required.

2.2.3 WORK LIGHT

Operator control of machine's work light(s). Pressing push button toggles work light On/Off.
Machine work light is set to automatically turn on with power-up of machine.

2.2.4 ALARM MSG

Pressing this button will reset the CNC and clear the PMC message.

2.3 OPERATION MODE SELECTION

2.3.1 AUTO

Auto (Memory) Mode   (Fanuc operator manual GFZ-63534EN Section 4.1)

Auto Mode also is called Memory Mode. Automatic operation of part program selected from program files registered in control's program directory.

2.3.2 EDIT

Edit Mode   (Fanuc operator manual GFZ-63534EN, Section 9)

Edit mode enables entering and editing of part programs stored in control's part program directory. Part programs stored on optional Data Server or memory card inserted in PCMCIA card slot are not available for editing. Programs must be edited before loading to storage media.

2.3.3 MDI

Manual Data Input Mode   (Fanuc operator manual GFZ-63534EN, Section 4.2)

MDI mode is used for simple test operation. In the MDI mode, operator can create and execute a program consisting up to 10 lines from the MDI panel, which is in the same format as the normal program.

2.3.4 **REMOTE**   Remote Mode   (Fanuc operator manual GFZ-63534EN, Section 4.3, 4.13)

Remote mode is also called DNC mode. In the mode, it is possible to perform machining while a program is being read in via reader/puncher interface, or remote buffer. Operator can, also, perform machining with execution of the program in the memory card, which is installed in the memory card interface, which is in the left side of the screen.

2.3.5 **REF RETURNE**   Reference Return Mode   (Fanuc operator manual GFZ-63534EN, Section 3.1)

Reference Return gives an opportunity to return all the axes to the machine zero position.

2.3.6 **JOG**   Continuous Jog Mode   (Fanuc operator manual GFZ-63534EN, Section 3.3)

In the continuous jog mode, pressing the direction switch on the operator's panel moves the tool along with the selected axes in the selected direction.

2.3.7 **HANDLE**   Handle (MPG) Mode   (Fanuc operator manual GFZ-63534EN, Section 3.4)

In the handle mode, operator can move the axes by rotating the manual pulse generator on the operator's panel.

3.0 MANUAL OPERATION

3.1 MANUAL OPERA-TION MODES

3.1.1 MANUAL REFER-ENCE POSITION RETURN

Reference Zero:

Manual reference position return is to move the tool to the reference position using switches and push buttons located on the operator's panel. Fadal machines are equipped with Absolute Encoders. Stored position information from the encoders will initialize the current machine position at power-up. Manual referencing of axes after power-up should not be required. When position is established by absolute feedback, REF RETURN MODE will simply command axes to Home position.

Reference Procedure:

1. Select **REF RETURN** Mode

2. Select one axis to be referenced by **X, Y, Z, 4,** or **5** axes button.
Machine axes will move at traverse rate to home position. When each axis has completed finding the reference position, the related machine coordinate will become to zero. Press the JOG, JOG INC, HANDLE or AUTO, MDI, and EDIT mode keys to exit the Reference mode.
Pressing the POS key on the MDI panel will switch to different position display which includes machine coordinate, absolute coordinate, and related coordinate.
For referencing of machine axes, including initialization of absolute encoder positions, please see separate maintenance procedure in Appendix.

☞

> ***NOTE:***
> FADAL machines do not use reference deceleration switches for establishing machine home position. Dog-less reference operation is provided for referencing axes without reference switches. Maintenance instructions for dog-less reference must be followed exactly, otherwise machine home position will not be repeatable.

3.1.2 **JOG OPERATION**

1. Select **JOG** Mode.

2. Select one axis to be manually jogged by **X, Y, Z, 4** or **5** axes buttons. Axis selection button does not have to be held down. LED will light indicating selected axis. Pressing any axis button will automatically de-select previously selected axis. It is implemented to prevent jogging of more than one axis simultaneously.

3. Press and hold **+** or **-** direction button for desired direction of travel. Axis motion will continue until direction button is released. Direction LED will be lit during motion.

Machine axes will travel at jog rate 200 inch per minute. Feedrate Override rotary switch may be used to vary jog rate.

4. Pressing **TRVRS** during continuous jog move will increase axis jog rate to traverse speed (MAX 900IPM for XY axis, 700IPM for Z axis). Feed rate rotary switch is also used to override traverse rates.

☞ **NOTE:**
   If the feedrate is at 0%, no motion will occur when using the "+/-" jog button.
   6030, 8030 have the Max 400ZPM for X. Y. Z.

3.1.3 INC JOG OPERATION
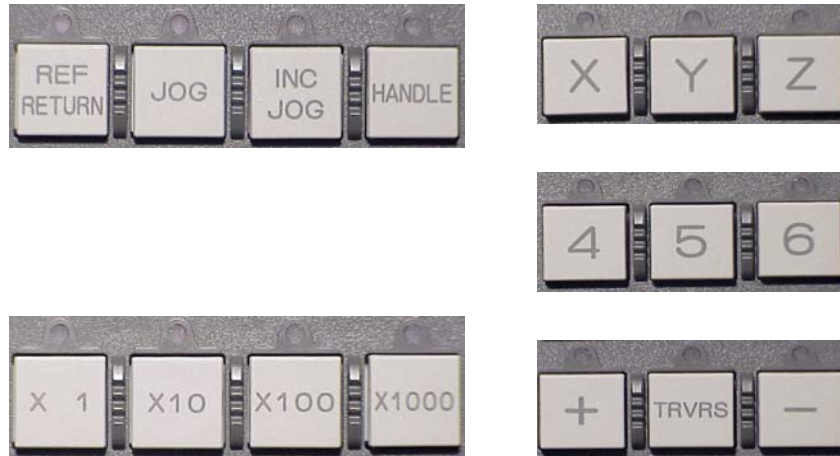
Each time a **+/-** button is pressed, the tool moves by the predetermined distance.

1. Select **INCR JOG** Mode.

2. Select one axis by **X, Y, Z, 4** or **5** axes buttons, as in JOG mode.

3. Select desired increment of travel by using **X1, X10, X100** or **X1000** push buttons.

4. Press **+** or **-** direction button for desired direction of travel. Incremental axis motion of selected distance will be commanded per each press of direction button. Direction LED will be lit during motion.

Incremental travel selection by multiplier setting (X1, X10, X100 or X1000) is based on the minimum command increment used by FADAL machine, 0.0001 inch or 0.001 mm. Selection of X1 will command 0.0001" or 0.001mm for each press of direction button. X10 commanding 0.001" or 0.01mm, (10 x 0.0001" or 10 x 0.001mm) and so on.

X1:   0.0001 inch (0.001mm)
X10:   0.001 inch (0.01mm)
X100:   0.01 inch (0.1mm)
X1000: 0.1 inch (1.0mm)

3.1.4 HANDLE OPERATION

Handle mode is also referred to as MPG Mode (Manual Pulse Generator). By rotating the manual handle, the tool moves by the distance corresponding to the degree of handle rotation.

1. Select **Handle** Mode.

2. Select one axis by **X, Y, Z, 4** or **5** axes buttons, as in JOG mode.

3. Select desired increment of travel by using **X1, X10, X100** or **X1000** push buttons.

   X1:   0.0001 inch (0.001mm)
   X10:   0.001 inch (0.01mm)
   X100: 0.01 inch (0.1mm)

4. Rotate manual pulse generator (MPG) clock-wise or counter clock-wise for plus or minus motion, respectively. For each detent or "click" of MPG handle, one selected increment or travel will be commanded. Direction LED will be lit during motion.

☞ **NOTE:**
   Increment travel selection by multiplier settings are used same as during INCR JOG mode. For example, selection of X10 increment with clock-wise rotation of 15 "clicks" on MPG will command 0.0150" or 0.150mm of travel in plus direction.

☞ **NOTE:**
   When MPG hand wheel is rotated at a rate fast enough to exceed the axis traverse rate, the axis speed is claimped at the traverse rate and excess rotation of the hand wheel is ignored. In this case, the distance of axis travel will not equal the amount of hand wheel rotation.

3.1.5 **JOG FEED RATE**

When the machine is in the JOG mode, the jog feedrate varies from 0 IPM to 240 IPM since the feedrate switch changes the feedrate from 0% to120%. For example, if Jog Feedrate Switch is at 100%, the feedrate changes to 200IPM.
This switch affects the rapid speed since it is rapid speed override switch also.
When the HMOP operation ON/OFF switch is turn off, the effective feed override/ feedrate is only related with main operator panel feed override switch.
When the operation ON/OFF switch is turned on, the effective feed override/feedrate value is that HMOP override switch's value multiply main operator panel feed override switch's value.

☞ **NOTE:**
   If the feed override/feed rate is at 0%, no motion will occur when using the "+/-" jog button or executing the program.

4.0 AUTOMATIC OPERATION

4.1 AUTOMATIC OPER-ATION MODES

4.1.1 MDI OPERATION

In the MDI mode, a program consisting of up to 10 lines can be created in the same format as normal programs and executed from the MDI panel. MDI operation is used for simple test operations.
In order to complete this operation, follow the next procedure:

1. Press the MDI mode selection switch.

2. Press the **PROG** function key on the MDI panel to select the program screen.

3. Prepare a program to be executed by an operation similar to normal program editing. M30, specified in the last block can return control to the beginning of the program after operation ends. Word insertion, modification, deletion, word search, address search, and program search are available for programs created in the MDI mode.

4. To entirely erase created program in the MDI mode, use one of the following methods:
    • Enter address, and then press the **DELETE** key on the MDI panel.

    • Alternatively, press the **RESET** key.

5. To execute a program, set the cursor on the head of the program. Push **CYCLE START** button on the operator's panel. By this selection, the prepared program will start. When the program end (M02, M30) or ER (%) is executed, the program will be erased and the operation will end. By command of M30, control returns to the head of the prepared program.

6. To stop or terminate MDI operation in midway through, follow the next steps:
    • Stop MDI operation.
    Press the feed hold switch on the machine operator's panel. The feed hold **LED** goes on and the cycle start **LED** goes off.

    • Terminate MDI operation.
    Press the reset key on the MDI panel. Automatic operation is terminated and the reset state is entered. When a reset is applied during movement, movement decelerates then stops.

4.1.2 MEMORY OPERATION (AUTO MODE)

Programs are registered in memory in advance. When one on these programs is selected and the cycle start switch on the machine operator's panel is pressed, automatic operation starts, and the cycle start **LED** goes on.

When the feed hold switch in the machine operator's panel is pressed during automatic operation, this operation is stopped temporarily. When the cycle start switch is pressed again, automatic operation is restarted.

When the **RESET** key on the MDI panel is pressed, automatic operation terminates and the reset state is entered.

To complete this operation follow the next procedure:

1. Press the **MEMORY (AUTO)** mode selection switch.

2. Select the program from the registered programs doing the following steps:

   • Press **PROG** to display the program screen.

   • Press **"+"** soft key.

   • Press **"DIR"** soft key and the program library will be displayed.

   • Enter a program number using the numeric keys.

   • Press the **O SRH** soft key.

3. Press the cycle start switch on the machine operator's panel. Automatic operation starts, and the cycle start **LED** goes on. When automatic operation terminates, the cycle start **LED** goes off.

4. To stop or cancel memory operation midway through, follow the steps below:

   • Stopping memory operation.

   Press the feed hold switch on the machine operator's panel. The feed hold **LED** goes on and the cycle start **LED** goes off. The machine responds as follows:
   When the machine was moving, feed operation decelerates and stops.
   When dwell was being performed, dwell is stopped.
   When M, S, or T was being executed, the operation stopped after M, S, or T is finished.

   • Terminating memory operation.

   Press the **RESET** key on the MDI panel.
   Automatic operation is terminated and the reset state is entered. When a reset is applied during movement, movement decelerates then stops.

FADAL MACHINING CENTERS

4.1.3 DNC OPERATION

By activating automatic operation during the DNC operation mode **(REMOTE)**, it is possible to perform machining (DNC operation) while a program is being read in via reader/puncher interface, or remote buffer. If the floppy cassette directory display option is available, it is possible to select files (programs) saved in an external input/output unit of a floppy format (Handy file, Floppy Cassettes, or FA card) and specify (schedule) the sequence and frequency of execution for automatic operation.

To use the DNC operation function it is necessary to set the parameters related to the reader/punch interface and remote buffer in advance.

To complete this procedure follow the next procedure:

1. Search for the program to be executed.

2. Press the **REMOTE** switch on the machine operator's panel to set **REMOTE** mode, then press cycle start switch. The selected file is executed.

During DNC operation, the program currently being executed is displayed on the program check screen and program screen.

The number of displayed program blocks depends on the program being executed.

DNC Operation procedure with memory card:
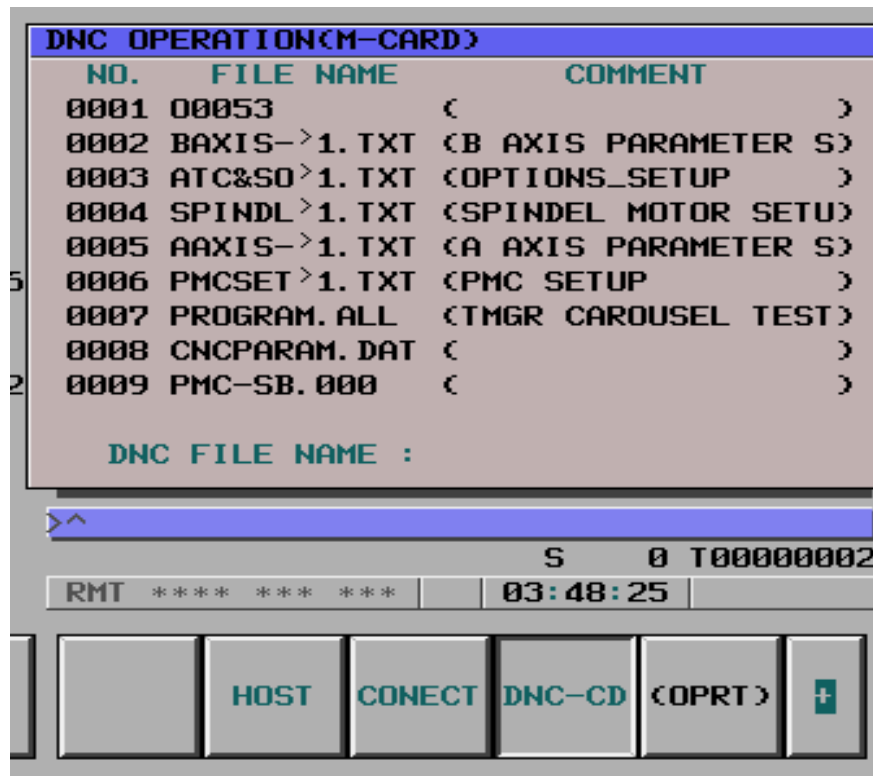
1. Set the parameter of No.0020 to 4 in the setting screen in advance.

2. Change to **REMOTE** mode.

3. Press **PROG** function key on the MDI panel.

4. Push **"+"** soft key twice.

5. When **"DNC-CD"** soft key is pressed, the following screen is displayed.

    The screen can be scrolled by page key. An arbitrary file number is input, and **"F SRH"** soft key is pressed. Then the arbitrary file name is displayed at the top of DNC operation (memory card) screen.

6. Input the file number which is going to be executed.

7. Press the **"DNC-ST"** soft key, the file name will be displayed in the right side of **DNC FILE NAME:**
    For example, DNC FILE NAME:O0053

8. Press the **CYCLE START** button to execute the program selected.

4.1.4 MEMORY PROTECTION KEY

This key is used to prevent part programs, offset values, parameters, and setting data from being registered, modified, or deleted erroneously.
When the key is in the On position **"I"**, the memory is protected and operator can't change any data in the CNC. When the key is in the Off position **"O"** operator is able to register/modify/delete the part programs, offset values, parameters and setting data.

FADAL MACHINING CENTERS

5.0 TEST FUNCTION

5.1 TEST FUNCTION MODES

5.1.1 SINGLE BLOCK

Single Block   (Fanuc Operator Manual GFZ-63534EN, III, Section 5.5)

Single Block Mode executes part program block by block. Single block mode is implemented to toggle On/Off with press of button.
Pressing the single block switch starts the single block mode. When the cycle start button is pressed in the single block mode, the tool stops after executing a single block in the program. Check the program in the single block mode by executing the program block by block.

5.1.2 BLOCK DELETE

Block Skip   (Fanuc Operator Manual GFZ-63534EN, II, Section 12.2)

Skip execution of program Block (/). Multi level Block Skip is not supported. Block Delete is implemented to toggle On/Off with press of button.

5.1.3 OPTION STOP

Optional Stop on M01   (Fanuc Operator Manual GFZ-63534EN, II, Section 11.1)

Option Stop is implemented to toggle On/Off with press of button. Executing program will stop at M01 when option stop button is On. Operator needs to push CYCLE START button to restart the program. It does not affect the program when the option stop button is Off

5.1.4 PROGRAM STOP

M00 or M01 Program Stop   (Fanuc Operator Manual GFZ-63534EN. III, Section 40.1)

Program Stop indicator LED is lit when part program execution is stopped by M00 or M01 part program.

5.1.5 MC LOCK

Machine Lock (Test Mode)   (Fanuc Operator Manual GFZ-63534EN, III, Section 5.1)

Machine Lock enables execution of part program without axis motion, but M/S/T command still is able to execute. This button is for test purposes. Machine lock is implemented to toggle On/Off with press of button.

5.1.6 **DRY RUN**

Dry Run Feed Rate   (Fanuc Operator Manual GFZ-63534EN, III, Section 5.4)

This feed rate forces program federate to fixed "dry run" rate to speed non-cutting testing of part programs. Dry Run is implemented to toggle On/Off with press of button. The tool is moved at the feed rate specified by a parameter regardless of the feed rate specified in the program. This function is used for checking the movement of the tool under the state that the workpiece is removed from the table. Press the Dry Run switch on the machine operator's panel during automatic operation. The tool moves at the feed rate 900/700 IPM when the feed rate is overridden by 100%. The rapid traverse switch can also be used for changing the feed rate. The Dry Run feed rate changes as shown in the table below according to the rapid traverse switch and parameters.

For X and Y axis, when the DRY RUN button is "ON".

Rapid traverse button	Program command	
	Rapid traverse	Feed
ON	700IPM	500IPM
OFF	900* Feedrate override IPM	500*Feedrate override IPM *1

For Z axis, when the DRY RUN button is "ON".

Rapid traverse button	Program command	
	Rapid traverse	Feed
ON	700IPM	500IPM
OFF	700* Feedrate override IPM	500*Feedrate override IPM *1

*1. Speed is Clamped to maximum feedrate 500IPM

☞ | **NOTE:**
When the machine is in the Auto/Remote mode, operator can only turn on Dry Run by pressing Dry Run & Func key at the same time.

6.0 MACHINE SETUP FUNCTION

6.1 SETUP MODES

6.1.1 WORK MEAS

Work Offset Setting   (Fanuc Operator Manual GFZ-63534EN, III, Section 11.4.14)

This measurement simplifies setup of work/fixture offsets using Measurement B feature. Please refer the Fanuc operator manual for the details.

6.1.2 TOOL MEAS

Tool Length   (Fanuc Operator Manual GFZ-63534EN, III, Section 11.4.14)

This measurement simplifies setup of tool length offsets using Measurement B feature. Please refer the Fanuc operator manual for the details.

6.1.3 FUNCTION

Function button **FUNC**

Horizontal mounting of Operator Panel in Fadal pendant exposes operator buttons to a more varied environment. To prevent accidental activation of selected operator panel operations, the **FUNC** button is required to be pressed in combination with desired operator panel function.
Operator Panel functions currently requiring **FUNC** button are:

HOME PSN	**LOAD PSN**	**Z HOME**	**INIT MAG**
TOOL DATA	**TOOL CHNG**	**LOAD TOOL**	
SPDL CCW	**SPDL CW**	**TOOL BRKN**	

6.1.4 HOME PSN

Macro Program Positioning Axes to Home Position   **FUNC**

Pressing this button will call the macro program which commands axes to machine position of X0.0 Y0.0 Z0.0 using G53 traverse moves. Program will command Z-axis to 0.0 first, before X & Y axes motion are commanded.
FUNC button must be pressed simultaneously with **HOME PSN** to execute program.

6.1.5 **LOAD PSN**

Macro Program Positioning Axes to Table Load PSN   **FUNC**

Pressing this button will call the macro program which commands axes to machine position of X0.0, Z0.0, and Y plus limit position using G53 traverse moves. Table position is in front and center. When control is set for metric operation, macro program will automatically command to X0.0, Y+Limit, Z0.0 in mm units **FUNC** button must be pressed simultaneously with **LOAD PSN** to execute program.

6.1.6 **Z HOME**

Macro Program Positioning Axes to Table Load PSN   **FUNC**

Pressing this button will call the macro program which commands Z-axis to machine position of Z0.0 using G53 traverse move.
FUNC button must be pressed simultaneously with **Z HOME** to execute program.

7.0 TOOL OPERATION

7.1 MANUAL OPERA-TION

7.1.1 DRUM FWD

Manual Index of Tool Drum in Forward Direction

Commands are manually jogging or indexing of the tool drum in forward direction. Forward direction is defined as indexing of tool pots in rising order:... 23, 24, 1, 2, 3,... Rotating is counter clock-wise when looking at the back of tool operator station.
Jog Mode is required. Operator may hold down button for indexing of multiple tool pots, or press momentarily to index one tool. **DRUM PSN** button can be used to display tool pot at tool change (bottom) position. Operator message will update while tool carousel is being manually rotated.

7.1.2 DRUM REV

Manual Index of Tool Drum in Reverse Direction

Commands are manually jogging or indexing of the tool drum in reverse direction. Reverse direction is defined as indexing of tool pots in decreasing order:... 3, 2, 1, 24, 23,... Rotating is clock-wise when looking at the back of tool drum from the operator station.
Jog Mode is required. Operation is same as used for **DRUM FWD**.

7.1.3 DRUM PSN

Display of Tool Drum Position by Operator Message

Use of this button activates display of Operator Message containing tool pocket number at tool change (button) position of carousel.
DRUM PSN message is set to automatically turn on with power-up of machine. Position of Tool Drum is maintained in battery backed PMC Data table while machine is powered down.
Operator may turn the message On/Off at any time using **DRUM PSN** push button.

7.1.4 TOOL REL

Tool Release (Draw Bar Open)

This button is for manually loading/uploading a tool holder. Operator must be prepared to catch tool from spindle cartridge immediately upon pressing button. Spindle air is turned on along with release of tool. Draw bar will open after pressing the button more that 1 second and remain open while push button is still held.
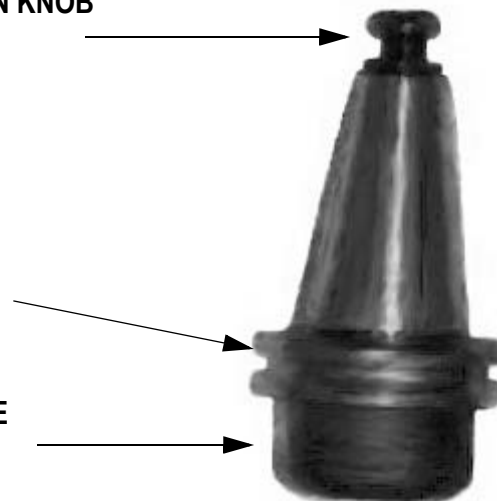Spindle must be stopped with control in JOG or HANDLE modes to enable tool release.

Unloading tool procedure:
1. The tool holder must be held in the left hand with the thumb and the first finger grasping the holder below "V" groove. No other fingers should have contact with the holder or the tool in the holder. The area below the "V" groove is called the safe zone. The safe zone is the only place where the tool holder should be held.

RETENTION KNOB

V GROOVE

SAFE ZONE

2. Press the **TOOL REL** button. Keep the **TOOL REL** button pressed until the tool is completely out of the spindle.

Loading tool procedure:
1. The tool holder must be held in the left hand with the thumb and the first finger grasping the holder below the "V" groove. No other fingers should have contact with the holder or the tool in the holder. The area bellow "V" groove is called the safe zone. The safe zone is the only place where the tool holder should be held.

2. Place the holder into the spindle after pressing the **TOOL REL** button, not before. The keys on the nose of the spindle must fit into the key-ways on the tool holder flange.

3. Release the **TOOL REL** button to lock the tool into the spindle.

☞ > **NOTE:**
> When loading a holder into the spindle, inspect the taper for chips and dents. Remove any chips or dents from the taper with a flat stone.
> Confirm the retention knob is securely tightened before placing the tool in the spindle.
> After load a tool holder info the spindle, operator needs to edit the magazine management to write the spindle tool number into the SPDL1 column, and then use the TOOL CHNG to load the spindle tool into the magazine/carousel.

7.1.5 TOOL CHNG

Macro Program Execute Tool Change   **FUNC**
(Only for **DATC**)

Pressing this push button will execute macro program O8988. Program executes swap of tool between spindle cartridge and current position of **TOOL DRUM**.
This button is for manually unloading of tools between spindle and tool drum, or loading of tool to specific tool drum pot. Operator can position tool drum prior to use of **TOOL CHNG** button.
During tool loading, operator may input tool number into spindle window on Tool Magazine screen. Tool Change cycle will move tool number to magazine. Alternatively, operator may edit magazine table once tools are loaded. Upon removal of tool from spindle, operator should delete tool number from spindle window on Tool Magazine screen.
FUNC button must be pressed simultaneously with **TOOL CHNG** to execute program.

7.1.6 TOOL LOAD

Macro Program for Loading Tools   **FUNC**

Machine with DATC:
Pressing this push button will execute the macro program O8989. Program searches for and positions to empty tool pot in tool drum. Then executes swap of new tool from spindle cartridge into empty tool pot.
This function is provided for manual tool loading from to tool drum. Operator to input tool number from spindle window on Tool Magazine screen so that tool number can be loaded to magazine, or edit magazine and spindle tool data after tools are loaded.
FUNC button must be pressed simultaneously with **LOAD TOOL** to execute program.

FUNC+LOAD TOOL procedure
- Manually load the tool to the spindle according the **TOOL REL** procedure.
- Press **JOG** or **HANDLE** mode.
- Press **OFFSET/SETTING** key on the MDI panel.
- Press **"+"** soft key below the LCD screen.

- Press **TL-MNG** soft key.
- Press **MAG** to enter the tool data management screen.
- Press **"OPRT"** soft key.
- Press **EDIT** soft key, the EDITING will be displayed on the screen.
- Move the cursor to the most right column: **SPDL1.**
- Input the tool number of the current spindle tool into the **Spindle Tool** column.
- Press **INPUT**, the tool number will be displayed in the **SPDL1** column.
- Press **FUNC+LOAD TOOL**, tool will be loading the magazine automatically, and the **MAG** management will be updated.
- Press **OPRT.**
- Press **END** to end the tool management editing.

☞
NOTE:
Operator need register the tool into the management first before load the new tool.

Machine with ATC:
Press the push button with **FUNC** button will execute the macro program O8989.
Program positions to specific tool pot in tool drum, and then load the spindle tool into the specific tool pot.
This function is provided for manually loading tools from spindle to **TOOL DRUM**.
Operator input tool number from spindle window on Tool Magazine screen so that tool number can be loaded to magazine, or edit magazine and spindle tool data after tools are loaded.
FUNC button must be pressed simultaneously with **LOAD TOOL** to execute program.

FUNC+LOAD TOOL procedure
- Manually load the tool to the spindle according the **TOOL REL** procedure.
- Press **JOG** or **HANDLE** mode.
- Press **OFFSET/SETTING** key on the MDI panel.
- Press **"+"** soft key below the LCD screen.
- Press **TL-MNG** soft key.
- Press **MAG** to enter the tool data management screen.
- Press **"OPRT"** soft key.
- Press **EDIT** soft key, the EDITING will be displayed on the screen.
- Move the cursor to the most right column: **SPDL1.**
- Input the tool number of the current spindle tool into the **Spindle Tool** column.
- Press **INPUT**, the tool number will be displayed in the **SPDL1** column.
- Press **FUNC+LOAD TOOL**, tool will be loading the magazine automatically, and the **MAG** management will be updated.
- Press **OPRT.**
- Press **END** to end the tool management editing.

☞
NOTE:
Operator need register the tool type into the management first before load the new tool.

☞
NOTE:
If the tool management screen, MAG has the same tool already, operator should clear this magazine pot using EDIT and take the tool out from that magazine manually first before try to load another tool.

7.1.7 REFERENCE THE DRUM (ONLY FOR DATC)

1. Press **REF RETURN** button.

2. Press **X1** button. The drum will automatically move to the number **1**.

7.2 AUTO OPERATION

7.2.1 T-COMMAND (TOOL NUMBER)

Eight digit T-Word may be programmed in block with or without M06 code.
T-Word programmed in block by itself will position tool carousel with position tool carousel with programmed tool at 6 o'clock position (DATC) for next tool change. Rotation of carousel will not inhibit continued execution of part program (FIN) as carousel is away from work area. For DATC, this enables cycle time reduction of part program execution, by allowing next tool to be immediately ready for tool change. Programming T-Word after M06 will command positioning of carousel, followed by immediate tool change. T-Word must be programmed following M06 in block, as T-Word is argument passed to M06 macro program.
Programming T-code of tool that is already at tool change (6 o'clock) position will command no activity by tool carousel, as requested tool, is already at tool change position.
T-Word may be used to position tool carousel by pocket number when combining with M100 miscellaneous code.

M100T5: Position carousel to tool pocket #5. Pocket #5 will be selected regardless of tool number in pocket, or if pocket empty.

7.2.2 TOOL CHANGER COMMAND

ATC:
M06 Txx command from part program executes tool change cycle for carousel type ATC tool changer. M06 command calls macro program O9021, which executes required Z-axis motion commands & special Miscellaneous Codes (M- Codes) for execution of the tool change cycle.
Tool number (T-Command) must always follow M06 code in program block. M06 programmed in block without T-Code following, or T-Code preceding M06 is illegal syntax for this machine.

M06 T10: Proper Command Syntax
M06:   Illegal, Macro Error 3020 will result
T05 M06: Illegal, Macro Error 3020 will result

☞ ***NOTE:***
Tool changer syntax on other makes/models of machines using GE Fanuc or Fanuc controls may differ. Operation of syntax on this machine is as listed above, and not selectable.

WARNING !
1. Special M-Codes contained in macro program O9021 must never be used in end-user part programs without specific permission of Fadal Machining Centers. Any use other than in O9021 macro cycle is the responsibility of the machine user/operator/programmer. Machine damage can occur.

2. End-user should always refer to Fadal Machining Centers Operator Manual: applicable to this machine, for proper M-Codes to be used during part programming.

3. Macro program O9021 must never be edited without permission of Fadal Machining Centers. Machine damage can occur.

DATC:
M06 command from part program executes tool change cycle for "dual arm" type DATC tool changer. M06 command calls program O9020, which executes required Z-axis motion commands & special Miscellaneous Codes (M-Codes) for execution of the tool change cycle.
Tool number (T-Command) must follow M06 code in program block, when M06 and Txx code are programmed in the same block. T-Code preceding M06 in same block is improper syntax for this machine. T-Code may be programmed alone, in previous block to provide positioning of TOOL DRUM prior to tool change. This method improves part program cycle time.

T10:   *Tool Drum positions, following operations will continue*
                  while drum.
G01 X . . .,   *Positions to Tool from pocket containing T10 tool.*
. . . ,
. . . ,
M06:   *Tool change. T10 will be inserted into spindle, old tool to*
                  empty pocket. D and/or H codes must be added after tool
                  change for offsets as required.
M06:T4   *Legal. Tool Drum will position during tool change.*
T05M06:   *Illegal command. Tool changes as intended will not occur.*

Programming M06, without programming a new tool number, will not command a swap of tools between the carousel and spindle. When no new tool number has been specified, and the active tool is already in the spindle, it is expected that no tool change is required.

☞ *NOTE:*
> Tool changer syntax on other makes/models of machines using GE Fanuc or Fanuc controls may differ. Operation of syntax on this machine is as listed above, and not otherwise selectable.

✋ *WARNING !*

1. Special M-Codes contained in macro program O9020 must never be used in end-user part programs without specific permission of Fadal Machining Centers. Any use other than in O9020 macro cycle is the responsibility of the machine user/operator/programmer. Machine damage can occur.

2. End-user should always refer to Fadal Machining Centers Operator Manual: applicable to this machine, for proper M-Codes to be used during part programming.

3. Macro program O9020 must never be edited without permission of Fadal Machining Centers. Machine Damage could occur.

7.3 TOOL MANAGEMENT

7.3.1 INIT MAG

Macro Program Initial Tool Magazine Data   **FUNC**

Pressing this push button will execute macro program O8986 which initials tool data from magazine. Tool numbers in all tool drum pots are initialized to the related pocket number. Tool number in the spindle is initialized to "0" indicating no tool in the spindle. So, operator should take the tool off from the spindle after execution of this function. **FUNC** button must be pressed simultaneously with **INIT/CLR MAG** to execute program.

☞ *NOTE:*
> Please take off the spindle tool after execute this function since this function clears the spindle tool number to zero. Ignore this note to take off the tool from the spindle will cause tool changer crash.

7.3.2 TOOL DATA

Macro Program Loads Tool Management Data   **FUNC**

Pressing this push button will execute macro program O8987 which demonstrates loading of tool management data by part program command. Tool numbers 1-24 (1-maximum of tool pocket numbers) are loaded as well as tool groups demonstrating tool life setting by count and by time. The default setting is not tool management for those tools of this function. **FUNC** button must be pressed simultaneously with **TOOL DATA** to execute program.

☞ *NOTE:*
> User may setup the tool management function by manually setup the tool management screen according their requirement.

| 7.3.3 **TOOL BRKN** | Tool Broken | **FUNC** |

Pressing **TOOL BRKN** and **FUNC** button at the same time is a shortcut to mark active tool as broken/ damaged in the tool management. This function enables operator to mark tool as damaged without editing Tool Management Data. Tool management will not select a damaged tool for use, from the tool group.

7.3.4 **EDIT THE TOOL/ MAGAZINE LIST**

Edit the tool data:
- Press **MDI** button.
- Press **OFFSET/ SETTING**.
- Press **"+"** soft key.
- Press **TL-MNG** soft key.
- Press **TOOL** soft key to enter the tool data page.
- Press **OPRT** soft key.
- Press **EDIT** soft key to start editing the tool data.
- Press **OPRT** soft key gain after editing is finished.
- Press **END** soft key to exit tool data the editing mode.

Edit the Magazine data:
- Press **MDI** button.
- Press **OFFSET/SETTING.**
- Press **"+"** soft key.
- Press **TL-MNG** soft key.
- Press **MAG** soft key to enter the magazine data page.
- Press **OPRT** soft key.
- Press **EDIT** soft key to start editing the magazine data.
- Press **OPRT** soft key again after editing is finished.
- Press **END** soft key to exit magazine data the editing mode.

7.3.5 **TOOL ORDER PROCEDURE (ONLY FOR ATC)**

Procedure:
1. Move the drum to the number 1.

2. Execute the M11 to establish the tool order (the pocket number1) in the **MDI** mode.

3. Cycle the power.

☞ **NOTE:**
> If user does not cycle the power after M11, that may cause the tool changer crash happening.

7.3.6 **TOOL LOADING (ATC)**

Turret (ATC) open for loading:
1. Change the machine to the **MDI** mode and execute the M150.

M150 M-Code is provided to emulate Fadal control operation for manual tool change. M150 will return tool in the spindle to carousel pocket and raise Z-axis to clear tools. Carousel in left "out" position, enables operator to rotate carousel to next tool.

☞
> **NOTE:**
>
> M151 command may be used to pick up next tool, if desired. M150 will position above carousel when no tool is in the spindle to be returned.

2. Change the mode to Jog, and use the **DRUMFWD/DRUM REV** button to rotate the turret.
   Operator can press **FUNC+DRUM POS** button to display the drum current position.

3. Change the machine to the **MDI** mode and execute the M151.
   M151 M-Code is provided to emulate Fadal control operation for manual tool change. M151 will proceed from position machine left by M150 cycle. After operator has manually indexed tool carousel to desired new tool, executing M151 will pick up operator-selected tool from carousel and return carousel to home position.

☞
> **NOTE:**
>
> When user loads tools into the turret, they can place tools in the turret and use DRUM FWR/DRUM REV button to put the next tool in the turret until all the tools are placed in the turret. Rotate the turret until the first tool is in the number one position before setting the tool order (using M11 to establish tool order).

☞
> **WARNING !**
>
> 1. M150 is provided for MDI use only to return tool to carousel. Use of the M150/M151 commands is responsibility of the machine operator. Machine damage may occur in unintended use.
>
> 2. M151 is provided for MDI use only after M150 command for return of active tool to carousel, and positioning of Z-axis above the carousel. Use of M150/M151 is the responsibility of the machine operator. Machine damage can occur during unintended use.
>
> 3. M150/M151 commands should never be used during Automatic execution of part programs.

7.3.7 TOOL CHANGER (RECOVERY ATC)

1. Press the RESET to disable the alarm.

☞
> **NOTE:**
>
> If the alarm can not be disable, operator may cycle the power to clear the alarms.

2. Execute the M186 in the MDI mode to get the turret to the home position if the Z axis still is in the home position.

☞ | **NOTE:**
User may move the Z axis to the safe area before return of the slide according to the real situation.

3. Move to the cold start position using either REF RETURN or G28.

4. Use the DRUM FWD/DRUM REV button to rotate the turret to the number 1.

5. Execute the M11 to establish the drum number 1. Cycle the power.

6. Double check the tool management to see if it is right. Edit it to make it right if it needs.

8.0 COOLANT OPERATION

8.1 COOLANT CON-TROL PUSH BUTTONS
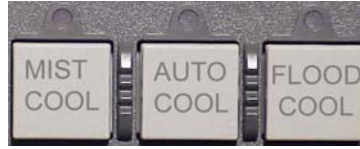
8.1.1 AUTO COOL

AUTO COOL push button selects automatic control of coolant by programmed M-Code.

AUTO COOL button toggles on/off, as indicated by LED. While AUTO COOL is active, manual coolant control buttons MIST COOL AND FLOOD COOL are inhibited. AUTO COOL "OFF" enables control of coolant. AUTO COOL is defaulted to "ON" condition at power-up.

When AUTO COOL is on, the following M-Code is used to control the coolant system:

M-Code:

M7: Mist coolant ON; Coolant though spindle ON
M8: Flood coolant ON
M9: Mist coolant, Flood coolant, coolant though spindle OFF

8.1.2 FLOOD COOL

FLOOD COOL turns on/off the flood coolant pump when the AUTO COOL is off.

8.1.3 MIST COOL

MIST COOL turns on/off the mist coolant pump and coolant through spindle solenoid when the AUTO COOL is off.

☞ **NOTE:**
FLOOD COOL and MIST COOL may be turned on simultaneously. MIST COOL also turns on the coolant through spindle pump if the machine has this option.

9.0 SPINDLE OPERATION

9.1 SPINDLE OPERA-TION PUSH BUTTONS

9.1.1 SPDL STOP

Manual Spindle Stop

SPDL STOP applies stop to spindle running in either CW or CCW direction. SPDL STOP button is inhibited when CYCLE START LED is on, indicating automatic cycle is active. SPDL STOP does not cancel active spindle speed "S" word command. SPDL STOP may be used from any automatic or manual mode.

9.1.2 SPDL CW

Manual Spindle Start Clock-Wise & Spindle Jog CW   **FUNC**

SPDL SW provides dual functions, based on operating mode of control.

9.1.3 SPDL CCW

Manual Spindle Start Counter Clock-Wise & Spindle Jog CCW   **FUNC**

SPDL CCW, also, provides dual function, based on operating mode of control.

	JOG Mode	MDI, AUTO or REMOTE Mode	
SPDL CW	X		Jog Spindle in CW direction
		X	Spindle Start in CW direction
SPDL CCW	X		Jog Spindle in CCW direction
		X	Spindle Start in CCW direction

9.1.4 MANUAL SPINDLE SPEED

Spindle jog speed is defined in PMC data table. Spindle Speed Override rotary switch is applied to spindle jog speed. Spindle will jog while FUNC+ SPNDL CW/CCW button is held. Spindle will stop when CW/CCW button is released.

9.1.5 M CODE / S CODE

M command:

> *M3:   Spindle CW rotate.*
> *M4:   Spindle CWW rotate.*
> *M5:   Spindle Stop.*
> *M41: Spindle low gear range.*
> *M42: Spindle high gear range.*

S command: Spindle rotation speed command

The spindle speed can be specified directly by address S followed by a maximum five-digit value (min-1). The unit is rotation per minute (RPM).

The S command must be specified with M3/M4 command in the same block to run the spindle. Using M5 command or pressing RESET can stop the spindle.

> Example: S10000; spindle rotation speed is 10000RPM.
> *M3 S2000 (Spindle CW, 2000RPM).*
> *M4 S2000 (Spindle CCW, 2000RPM).*
> *M5 Spindle stop.*

☞
> *NOTE:*
> Use of FUNC+SPDL CW/CCW as spindle start in MDI, Auto & Remote Modes is inhibited when Cycle Start LED is on, and also by state of door interlock. Programmed "S" word must be active for use of buttons for spindle start. Care should also be used in CW/CCW rotation.

9.1.6 SPINDLE SPEED RANGE

The spindle speed of Fadal VMC has the different configuration.

For EMCD, 2016, 2216, 3016 machines with maximum spindle speed 7.5K, the spindle has a single range30-7500 RPM.
For 2216, 3016 machines with maximum spindle speed 15K, the spindle has a single range 30-15000 RPM.
For 3020, 4020, 4525, 5020, 6030, 8030, 6535 40T machines with maximum spindle speed 15K with a electrical high/low, the spindle range is as follows:

Low range: 30-2500
High range: 2501-15000

The VMC machines with mechanical Hi/Lo and the maximum speed Is 10K, the spindle ranges are as follows:

Low range: 30-2500
High range: 2501-10000

9.1.7 **SPINDLE OVERRIDE**

Spindle override 50%-120%
This switch will adjust the commanded spindle speed by 50% to 120%.

9.1.8 **SPINDLE ORIENTATION**

Spindle orientation command for tool changer:

M190 : Spindle orientation
Execution of the M5/M3/M4 command or pressing the RESET key will release the spindle orientation command.

M19: Spindle special orientation command for cycles
> M19 positions spindle aligned with X & Y plane to provide for proper tool alignment during M19 orient cycles which is used during Fine Boring and Back boring canned cycles.

☞

NOTE:
> M19 should be used for the Fine Boring and Black Boring canned cycles only. Using M19 for other purposes may cause machine crash.

10.0 HMOP OPERATION GUIDE

SINGLE LED

EMERGENCY STOP

OPERATION ON/OFF

INPUT KEY WITH LEDx20

OVERRIDE SWITCH

MANUAL PULSE GENERATOR

CAUTION: (refer to FANUC HMOP manual)
Please hold the handy machine operator's panel through belt. When operator holds the handy machine operator's panel without using belt, the handy machine operator's panel may be dropped. This unit has an enabling switch. When operator is surprised with unexpected motion, he should release the enabling switch.

10.1 HMOP OPERA-TION ON/OFF SWITCH

Turn on this switch for the purpose of selecting the operation by handy machine operator's panel. Turn off this switch for operation by main operator's panel. When HMOP ON/OFF switch is turned on, the operator is permitted to use machine using HMOP only. If HMOP ON/Off switch is turned off, the operator is permitted to use machine using pendant.

CAUTION:
Please do not turn off the operation ON/OFF switch in the situation when operator exposes himself to danger. When the operation ON/OFF switch is turned off, the operation from main operator's panel is possible, but it is dangerous.

10.2 OVERRIDE SWITCH

The function of this switch is same as the feed rate switch on the main operator panel. When the operation ON/OFF switch is turned on, the effective override value is that HMOP override switch's value multiply main operator panel feed override switch's value.

Example: The operation switch is on.
Main operator panel feed override is 80%
HMOP feed override is 50%
The effective feed override for the machine is 40%
80% x 50% = 40%

When the operation ON/OFF switch is turned off, the effective feed override is only related with main operator panel feed override switch.

The keys for each of the next procedures are shaded.

10.3 MEMORY OPERATION PROCEDURE

XYZ 456	REF	SPINDL START	SPINDL STOP	RESET
HANDLE	JOG	SPINDL INC	SPINDL DEC	+
Xn	MEM	TOOL REL(F1)	MESSAGE	RAPID
SINGLE BLOCK	DRY RUN	CYCLE START	FEED HOLD	−

• Refer the FANUC operator manual to select the program and call it out to memory to run.
• Turn on the HMOP operation ON/OFF switch.

• When the **[MEM]** key is pushed, mode is changed to MEM operation mode.

• When the **[CYCLE START]** key is pushed, automatic operation is started.

• When the **[FEED HOLD]** key is pushed, automatic operation is stopped.
• When the **[RESET]** key is pushed, CNC is reset. Reset and rewind signal should be on for rewinding part program.

• Override can be applied to feed rate speed by the override switch on handy machine operator's panel. Set feed rate override signal from 0% to 120% according to the override switch position.

• For test operation **[DRY RUN]** and **[SINGLE BLOCK]** are available.

• One axis position is displayed normally. When the **[XYZ456]** key is pushed next axis position is displayed.

10.4 MANUAL HANDLE FEED PROCEDURE

XYZ 456	REF	SPINDL START	SPINDL STOP	RESET
HANDLE	JOG	SPINDL INC	SPINDL DEC	+
Xn	MEM	TOOL REL(F1)	MESSAGE	RAPID
SINGLE BLOCK	DRY RUN	CYCLE START	FEED HOLD	-

• When the **[HANDLE]** key is pushed, mode is changed to manual handle feed mode.

• The axis, which is moved by MPG, is selected by pushing the **[XYZ456]** key several times.

• When **[Xn]** key is pushed, manual handle feed amount selection is applied. Please select **X1-X10-X100** by pressing several times of this key.

• When the **[RESET]** key is pushed, the moving axis is stopped.

10.5 MANUAL JOG FEED PROCEDURE

XYZ 456	REF	SPINDL START	SPINDL STOP	RESET
HANDLE	JOG	SPINDL INC	SPINDL DEC	+
Xn	MEM	TOOL REL(F1)	MESSAGE	RAPID
SINGLE BLOCK	DRY RUN	CYCLE START	FEED HOLD	-

• When the **[JOG]** key is pushed, mode is changed to manual continuous feed mode.

• The axis, which is moved by the **[JOG]** key, is selected by pushing the **[XYZ456]** key several times.

• When the **"+"** key is pushed, the desired axis moves to positive direction. When the **"-"** key is pushed, the desired axis moves to negative direction.

• When the **[RAPID]** key is pushed, at the same time, rapid traverse is selected.

• Override can be applied to manual feed rate by the override switch on the handy machine operator's panel. Set manual feed rate override signal from 0% to 120% (0 IPM to 240 IPM) according to the override switch position.

• When the **[RESET]** key is pushed, the moving axis are stopped.

10.6 MANUAL REFERENCE RETURN

XYZ 456	REF	SPINDL START	SPINDL STOP	RESET
HANDLE	JOG	SPINDL INC	SPINDL DEC	+
Xn	MEM	TOOL REL(F1)	MESSAGE	RAPID
SINGLE BLOCK	DRY RUN	CYCLE START	FEED HOLD	-

• When the **[REF]** key is pushed, mode is changed to manual continuous feed mode and manual reference position return selection signal "ZRN" is asserted.

• The axis, which is moved for manual reference position return, is selected by pushing the **[XYZ456]** key several times.

• When the **"+"** key or the **"-"** key is pushed, the axis moves to the direction of reference position.

10.7 MANUAL SPINDLE CONTROL

| XYZ 456	REF	SPINDL START	SPINDL STOP	RESET
HADNLE	JOG	SPINDL INC	SPINDL DEC	+
Xn	MEM	TOOL REL(F1)	MESSAGE	RAPID
SINGLE BLOCK	DRY RUN	CYCLE START	FEED HOLD	-

• When **[SPINDL START]** key is pushed, spindle motor is rotated to the direction which is designed with the speed at the decided override.

• When **[SPINDL STOP]** key is pushed, spindle motor is stopped.

• When **[SPINDL INC]** key or the "SPINDL DEC" key is pushed, override of speed is changed. The changing rate is 50% - 120%.

10.8 KEY SHEET LAYOUT

FANUC standard key sheet A is as follows. This key sheet is put on the standard unit.

Tool Release Button (F1 Function)

10.8.1 **TOOL REL**

XYZ 456	REF	SPINDL START	SPINDL STOP	RESET
HADNLE	JOG	SPINDL INC	SPINDL DEC	+
Xn	MEM	TOOL REL(F1)	MESSAGE	RAPID
SINGLE BLOCK	DRY RUN	CYCLE START	FEED HOLD	-

Same function as the TOOL REL button on the operator panel.
It is used for manually loading/unloading a tool holder use HMOP when the HMOP switch is on. Operator must be prepared to catch tool from spindle cartridge immediately upon pressing button. Spindle air is turned on along with release of tool. Draw Bar will open after pressing the button more than 1 second and remain open while push button is still held. Spindle must be stopped, with control in JPG or HANDLE modes to enable tool release.

☞ **NOTE:**
    Please refer the operator panel A section for more detail about release function.

10.9 **MESSAGE KEY**

When the "MESSAGE" key is pressed, the screen is changed to user message screen. The next message screen is displayed after another pressing the "MESSAGE" key. When the "XYZ456" key is pressed, the screen returns to the screen according to each mode.

FADAL MACHINING CENTERS

11.0 APPENDIX

11.1 **M CODE LIST**

M Command	Description	Remark
M00	Program Stop	See Unit II, Section 11.1 of 16i/18i Operator's Manual
M01	Optional Program Stop	See Unit II, Section 11.1 of 16i/18i Operator's Manual
M02	End-of-Program	See Unit II, Section 11.1 of 16i/18i Operator's Manual
M30	End-of-Program	See Unit II, Section 11.1 of 16i/18i Operator's Manual
M98	Sub-Program Call	See Unit II, Section 12.3 of 16i/18i Operator's Manual
M99	Sub Program End	See Unit II, Section 12.3 of 16i/18i Operator's Manual
M03	Spindle Start Forward	
M04	Spindle Start Reverse	
M05	Spindle Stop	
M06	Run Tool Change Cycle , M6 Txx	Macro Call O9020(DATC). Pass Tool number in macro call
		Macro Call O9021(ATC). Pass Tool number in macro call
M07	Coolant ON - Mist Coolant/Coolant Thru Spindle	Mist Coolant (Coolant 2) Receptacle
M08	Coolant ON - Flood Colant	
M09	Coolant OFF	
M11	Set Tool Carousel position to 1 (ATC Tool Changer ONLY)	Need to take off the tool from spindle and cycle the power
M19	FANUC canned cycle positioning, (Not for spindle orientation)	Canned Cycles, spindle orients normal to X/Y axes
M20	Wash Down, Toggle ON/OFF	option
M20	MIDACO pallet change 1	When the machine the MIDACO pallet option
M20	Does nothing	Standard.
M22	Chip Conveyor Toggle ON/OFF	option
M29	Rigid Tap	M29 Sxxxx in block prior to G84
M41	Low Gear Select	
M42	High Gear Select	
M48	100% Spindle Speed Override Forced	Spindle speed will override selection.
M49	100% Spindle Speed Override Released	
M60	A Axis Brake ON	

M61	A Axis Brake OFF	
M62	B Axis Brake ON	
M63	B Axis Brake OFF	
M64	M64/65 Output ON	
M65	M64/65 Output OFF	
M66	M66/67 Output ON, Chip Conveyor ON	
M67	M66/67 Output OFF, Chip Conveyor ON	
M68	M68/69 Output ON, Wash Down ON	
M69	M68/69 Output OFF, Wash Down OFF	
M78	M78/79 Output ON, Cooled spindle ON(2016 & 3016L)	option
M79	M78/79 Output OFF, Cooled spindle OFF(2016 & 3016L)	option
M80	MIDACO Pallet Changer 2	When the machine has MIDACO pallet option
M85	A Axis Rotary Table Enable	
M86	A Axis Rotary Table Disable	
M87	B Axis Rotary Table Enable	
M88	B Axis Rotary Table Disable	
M100	Position Tool Drum to Pocket by T-Word Ex: "M100 T05"	
M150	Return the current tool to ATC carousel(ATC ONLY)	Only use it with M151
M151	Tranfer the tool from carousel into spindle.(ATC ONLY)	Only use it with M150
M190	Spindle Orient for tool changer	Orient Tool to angle required for Tool Change

11.2 **ALARM LIST**

1. If it is a alarm related with CNC, please refer the CNC operator manual appendix.
2. Machine alarm lists.

Table 1: Alarm list

ALARM NO.	DESCRIPTION
2004	Operator function inhibited/check door overrides. alarm in cycle...
2005	Machine door opens. Feed hold on cycle start is inhibited
2006	Door override is active
2007	Close machine door to continue or reset to cancel operation
2010	Press cycle start to restart spindle
2011	Press cycle start to resume program command
2020	Low lube oil level refill vactra-2 or equivalent
2026	Air-oil fault, illegal table data for air-oil intervals
2027	Air-oil fault feed hold/spindle stop forced. See other messages
2030	Air-oil low oil pressure fault
2031	Air-oil upper bearing pressure failed fault
2032	Air-oil lower bearing pressure failed fault
2033	Air-oil high oil pressure while pump turned off
2034	Air-oil fault. Safety stop machine or stop will be forced
2035	Air-oil spindle air/vac pressure failed fault
2036	Air-oil level low. Please, refill reservoir
2037	Correct fault & use alarm -MSG PBTO all air-oil messages
2043	Chip conveyor overload trip maintenance must reset overload
2044	Tool arm overload trip maintenance must reset overload
2045	Tool magazine overload trip maintenance must reset overload
2046	Washdown pump overload trip maintenance must reset over-load
1050	Initial shift fail, CYC CTRT inhibit

Table 1: (Continued) Alarm list

ALARM NO.	DESCRIPTION
1051	Lost low gear during operation
1052	Lost HI gear during operation
1054	High to low gear shift failed
1055	Low to high gear shift failed
1083	Tool up at machine on failed
1084	Tool up at drum rotation failed
1086	Stored tool illegal, see OPR MSG
1090	No empty tool pockets available
1091	Invalid tool MGMT data for search
1092	Optional tool MGMT search illegal
1093	Empty pot search illegal data
1094	Empty pot search invalid pot
1095	Tool MGMT search protected
1096	Tool changer initial fault. see msg
1097	No tool changer, cycle start inhibited
1104	ATC tool count switch missing
1110	ATC carousel extend failed
1111	ATC carousel retract failed
1112	ATC carousel e-stop alarm
1120	M85 window R/W data error
1132	5th axis brake on, command halted
1134	4th axis brake on command halted
1360	M84 LOGR CALIB data zero error
1361	M84 LOGR SPDL speed calc error

Table 1: (Continued) Alarm list

ALARM NO.	DESCRIPTION
1362	M84 low gear shift error
1363	M84 LOGR WINDR read sample error
1364	M84 LOGR ARBITRARY data error
1365	M84 LOGR REV count windr error
1366	M84 LOGR COUNT windr error
1367	M84 LOGR data out of range error
1370	M84 LOGR DISP ratio math error
1371	M84 LOGR ratio range error
1372	M84 LOGR WINDW par set error 1
1373	M84 LOGR WINDW par set error 2
1374	M84 LOGR CALIB spindle start error
1376	M84 belt calib cycle inhibited
1380	M84 high calib data zero error
1381	M84 high spindle calc error
1382	M84 high gear shift error
1383	M84 high windw read sample error
1384	M84 high arbitrary data error
1385	M84 high rev count windw error
1386	M84 high FDBK count windw error
1387	M84 high data out of range error
1390	M84 high display ratio math error
1391	M84 high ration range error
1392	M84 high windw par set error 1
1393	M84 high calib spindle start error
1400	M84 #3741 calc error
1401	M84 #3742 calc error

Table 1: (Continued) Alarm list

ALARM NO.	DESCRIPTION
1402	M84 max low windw error
1403	M84 #3751 max hi window error
1404	M84 #3751 calc error
1405	M84 shift spindle window error
1406	M84 #3736 calc error
1407	M84 #3736 window error
1410	M84 #3752 window error
1411	M84 #3762 window error
1800	Adaptive feed hold on
1803	Adaptive alarm limit
1804	Adaptive PID error
1805	Adaptive speed error
1806	Adaptive command error
1807	Adaptive function error
2057	PWR down req'd to CLR alarm 1050. Check air pressure 80-90PSI
2060	Z axis tool change PROX switch not confirming tool change position
2061	TOOL DRUM pot not up for rotation of tool drum
2062	TOOL DRUM pot not in down position for access by tool arm
2063	TOOL ARM not in position for tool removal from spindle
2064	DRAWBAR not released for removal of tool from spindle
2065	TOOL ARM not in position for tool insertion into spindle
2066	DRAW BAR not clamped on new tool in spindle
2067	TOOL ARM not returned to park position at end of tool change

Table 1: (Continued) Alarm list

ALARM NO.	DESCRIPTION
2070	TOOL CHANGE command aborted tool change service mode active
2071	TOOL CHANGE command aborted spindle frwd, rev, orient active
2075	TOOL CHANGE & SPINDLE, e-stop forced alarm PB to clear
2076	Tool change service mode set
2077	Tool chng alarm, alarm PB to CLR cycle start inhibited
2081	Current ATC carousel position reset to pocket #1
2082	Tool drum at pocket #[I220, D500] press drum position to clear msg
2087	Stored tool value of 1-99999999 is not in legal range to #[I220,R6134]
2100	Axis inhibit by midaco pallet changer
2100	No tool changer selected correct K5.X & cycle power
2101	Illegal VMC tool changer setting correct K5.X & cycle power
2102	VMC & HMC tool changer both set correct K5.X & cycle power
2131	Use M61 to release 4th axis brake prior to axis command block
2131	Use M62 to release 5th axis brake prior to axis command block
2140	Adaptive warn limit
2377	M84 belt calibration active

11.3 **MACHINE COLD START PROCEDURE**

Fadal Fanuc VMC comes with absolute encoder which is setup in the Fadal already. The machine will remember the position even if it is powered off. We need to make the cold start position again if the alarm No.300 appears on the screen which can be caused by low battery or disconnect the servo motor encoder.

 Follow the next procedure:
1.   Prepare: In MDI mode input G21 to change machine to Metric display.

1.1 Press the MDI push button on the operator panel A.

1.2 Press PROGRAM key one/two times on the MDI panel until PROGRAM (MDI) screen is displayed.

1.3 Press G21 using MDI panel.

1.4 Press EOB key on the MDI panel.

1.5 Press INSERT key.

1.6 Press CYCLE START push button on the operator panel A.

1.7 Make sure the screen is changed to Metric Mode (X***,*** is displayed).

1.8 Press REF RETURN push button on the operator panel A.

1.9 Press Z push button on the operator panel A (Manual Z axis zero return).

1.10 Press X push button on the operator panel A (Manual X axis zero return).

1.11 Press Y push button on the operator panel A (Manual Y axis zero return).

1.12 Preparation is done.

2.   Reference all the axes.

2.1 Press REF RETURN and select reference mode, then push X/Y/Z key to move X/Y/Z axis to zero position. CNC screen will display X0Y0Z0.

3.   Install the tool.

3.1 Press JOG push button on the operator panel A.

3.2 Press TOOL REL push button on the operator panel and keep it. (Draw Bar will be released)

3.3 Put the tool inside of spindle nose.

3.4 Release the TOOL REL push button.

4. Find out X axis zero position manually, make the cold start.

4.1 Press HANDLE and select MPG mode, press X1/X10/X100 to select MPG resolution, and press X select to select X axis.

4.2 Use TAPE MEASURE find out the X axis middle point of the table, and make a mark.

4.3 Use MPG move the X axis to the X axis middle potion mark.

4.4 Record the X coordinate displayed on the CNC screen as A.

4.5 Press MDI push button on the operator panel A.

4.6 Press SYSTEM key on the MDI panel.

4.7 Press PARAMETER soft key below the LCD screen.

4.8 Press 1850 using MDI panel.

4.9 Press NO.SRH soft key below the LCD screen (Parameter 1850 will be displayed).

4.10 Move the cursor to X.

4.11 Press the X coordinator as following format: 34560 (3.456x10000).

4.12 Press INPUT key on the MDI panel (34560 will be displayed on the parameter 1850X column)
    ***Alarm 000 will pop up on the alarm window.
4.13 Power OFF the CNC (Press "O" button on the operator panel B).

4.14 Power ON the CNC (Press "T" button on the operator panel B).
    ***Alarm 300X will be displayed on the screen (it is normal).
4.15 Press JOG push button on the operator panel A.

4.16 Press X push button on the operator panel A.

4.17 Press "-" minus push button to move the X axis 55.0mm or more (press "+" push button if machine is 2016, 3016, EMCD, 2216).

4.18 Press "+" plus push button to move the X axis middle position (press "-" push button if machine is 2016, 3016, EMCD, 2216).

4.19 Press REF RETURN push button.

4.20 Press X push button. (X axis will automatically move to the zero position)

4.21 In case if zero position is wrong, repeat the previous steps starting with pressing MDI push button of the operator panel A.

4.22 Double check the cold start position using tape measure.

4.23 Press RESET key on the MDI panel. That will reset the 300 alarm.

5. Find out Y axis zero position manually, make the cold start.

    5.1 Install the indicator on the spindle.

    5.2 Repeat the same procedure as 4.1-4.23 to find out the Y axis cold start position.

6. Find out Z axis zero position manually, make the cold start.

    6.1 Install tool on the arm (DATC) or Magazine (ATC).

    6.2 Measure the height of the tool.

    6.3 Install the tool on the spindle.

    6.4 Use MPG move the Z axis to the same height as the tool on the ARM/Magazine.

    6.5 Repeat the same procedure as 4.5-1.23 to find out the Z axis cold start position.

7. Repeat the same procedure for A/B axis.

8. Cold Start setup is finished.

11.4 FANUC FILES BACKUP PROCEDURE

Please prepare the compact flash card.

Backup of data files procedure:
1. Power on VMC.
2. Place flash card in pendant's drive. (Located to left of pendant display).
3. Power on pendant "I" key.
4. Wait for pendant to boot up.
5. Press "System" key on pendant.
6. Press "Edit" key on pendant.
7. Soft key "+" twice.
8.
9. Soft key **"ALL IO"**.
10. Soft key "PRGRM".
11. Soft key "OPRT".
12. Soft key "PUNCH".
13. Enter file name "**1**".
14. Soft key "EXEC".   File called "1.)08504" should be created
15.
16. Soft key "<".

17. Soft key **"PARAM"**.
18. Soft key "OPRT".
19. Soft key "PUNCH".
20. Enter file name "**2**".
21. Soft key "EXEC".   File called "2.) CNCPARAM.DAT" should be created
22.
23. Soft key "<".
24. Soft key **"OFFSET"**.
25. Soft key "OPRT".
26. Soft key "PUNCH".
27. Enter file name "**3**".
28. Soft key "EXEC"   File called "3.) TOOLOFST.DAT" should be created
29.
30. Soft key "<".
31. Soft key "+" (next page).
32. Soft key **"MACRO"**.
33. Soft key "OPRT".
34. Soft key "PUNCH".
35. Enter file name "**4**".
36. Soft key "EXEC"   File called "4.) MACROVAR.DAT" should be created
37.
38. Soft key "<".
39. Soft key **"PITCH"**.
40. Soft key "OPRT".
41. Soft key "PUNCH".
42. Enter file name "**5**".
43. Soft key "EXEC"   File called "5.) PITCHERR.DAT" should be created
44.
45. Soft key "<".
46. Soft key **"WORK"**.
47. Soft key "OPRT".
48. Soft key "PUNCH".
49. Enter file name "**6**".   File called "6.) WORK-G54.DAT" should be created
50. Soft key "EXEC"
51.
52. Soft key "<".
53. Soft key "+" (next page).
54. Soft key **"TOOL"**.
55. Soft key "OPRT".
56. Soft key "PUNCH".
57. Enter file name "**7**".
58. Soft key "EXEC"   File called "7.) TOOL-MNG.DAT" should be created
59.
60. Soft key "<".
61. Soft key **"MAG"**.

62. Soft key "OPRT".
63. Soft key "PUNCH".
64. Enter file name "**8**".
65. Soft key "EXEC"   File called "8.) MAGAZINE.DAT" should be created
66.
67. Soft key "<".
68. Soft key **"CUSTOM"**.
69. Soft key "OPRT".
70. Soft key "PUNCH".
71. Enter file name "**9**".
72. Soft key "EXEC"   File called "9.) CUSTOMIZ.DAT" should be created
73.
74. Soft key "<".
75. Soft key **"STATUS"**.
76. Soft key "OPRT".
77. Soft key "PUNCH".
78. Enter file name "**10**".
79. Soft key "EXEC"   File called "10.) STATUS.DAT" should be created
80.
81. Soft key "<".
82. Soft key "+" five(5) times.
83. Soft key **"PMC"**.
84. Soft key "IO".
85. Soft key "M-CARD".
86. Soft key "WRITE".
87. Soft key **"LADDER"**.
88. File name should appear as "PMC-SB.000".
89. Soft key "EXEC".   File called "11.) PMC-SB.000" should be created
90.
91. Soft key "<".
92. Soft key "IO".
93. Soft key "M-CARD".
94. Soft key "WRITE".
95. Soft key **"PARAM"**.
96. File name should appear as "PMC-SB.PRM".
97. Soft key "EXEC".   File called "12.) PMC-SB.PRM" should be created
98.
99. Soft key "<" two (2) times.
100. Soft key **"SYSTEM"**.
101. Soft key "OPRT".
102. Soft key "PUNCH".
103. Soft key "EXEC".   File called "13.) SYS_CONF.DAT" should be created

12.0 INDEX